(12) UK Patent Application (19) GB (11) 2 227 887 (13) A

(43) Date of A publication 08.08.1990

(21) Application No 8830251.8

(22) Date of filing 24.12.1988

(71) Applicant
Technology Applications Company Limited

(Incorporated in the United Kingdom)

24 Ridgeway Street, Douglas, Isle of Man

(72) Inventors
John Michael Lowe
Joseph Devonport

(74) Agent and/or Address for Service
Eric Potter & Clarkson
St mary's Court, St Mary's Gate, Nottingham,
NG1 1LE, United Kingdom

(51) INT CL[5]
H05K 3/46 1/16

(52) UK CL (Edition K)
H1R RAK RAQ R10 R20
H1M MDX M1B2B7 M1B2C2 M1G6X M1G9A
U1S S1820 S2205

(56) Documents cited
GB 2157085 A GB 1401923 A GB 1385732 A
GB 1348659 A GB 1170991 A US 4118595 A

(58) Field of search
UK CL (Edition J) H1R RAK RAL RAQ
INT CL[4] H05K

(54) Making printed circuits

(57) A first conductive layer on a printed circuit board is separated from a superincumbent second layer by an insulating polymer layer. The arrangement may form a cross-over, a capacitor or a keyboard. The first conductive layer may be formed from a laminated copper foil by etching or may be screen printed using a conductive ink. Other methods such as transfer printing, lithography, air brush or hand brush may also be employed. The polymer layer and the second conductive layer may be deposited by screen printing. The conductive layers may be protected from corrosion by a carbon-containing ink which also acts as a solder resist during the attachment of other circuit components. Resistors may be deposited by screen printing, and the printed layers may be cured by heat or UV light. The invention may be used to make multi-layer circuits with up to 50 layers, including conductive ground layers to provide RFI shielding. The circuit substrate may be synthetic resin bonded paper, or the circuit may be formed directly on a car dashboard, radio case, heatsink or the like.

GB 2 227 887 A

This print takes account of replacement documents submitted after the date of filing to enable the application to comply with the formal requirements of the Patents Rules 1982.

IMPROVED METHOD FOR MAKING PRINTED CIRCUITS

BACKGROUND OF THE INVENTION

This invention is related to printed circuits for use in electrical and electronic equipment. In particular, it is an improved method of making printed-circuit boards and of producing printed circuits on surfaces other than conventional printed-circuit boards.

An element that is common to almost all electronic equipment is the printed circuit, typically in the form of a printed-circuit board (PCB). A PCB is generally made by laminating copper foil to a board. A desired pattern that includes conductors in the plane of at least one surface of the board is placed upon the board, and holes are drilled or punched for the mounting of components. This pattern is typically realized by placing a photosensitive resist on the board, exposing a photograph of the desired pattern on the resist, and developing the resist to produce a protective coating over the pattern. The remainder of the resist is treated to remove undeveloped resist from portions where it is desired to remove the copper. The

board is then treated with a process that removes the exposed copper. When the remaining resist is then removed, the desired pattern remains in the copper. Mounting holes for components are drilled or punched at desired locations somewhere in the course of this process, either before or after the removal step.

The materials most commonly used for PCBs are either polymerized epoxy materials containing glass fibers or paper bound by impregnated synthetic resins such as phenolic materials. The latter group is often referred to generically as synthetic-resin-bonded paper (SRBP). Boards made of either of these types of materials are clad with copper on one side or both sides, with the heat and pressure of the cladding or laminating process helping to cure the resin. The material cost of an SRBP PCB is typically about half as much as that of an epoxy fiberglass board of the same surface area, so there is a potential cost advantage when the SRBP board can be used. There are various bases for the selection between epoxy fiberglass and SRBP material. For example, the epoxy fiberglass boards are generally higher in strength and are preferred for use in equipment that may be subject to vibration.

A particular problem of circuit design that leads to complication in PCBs is the fact that not all electronic circuits can be made with their connections in a single plane. It is sometimes necessary to make

bridging connections between different portions of a circuit. A considerable amount of ingenuity goes into the design and layout of PCBs to minimize such bridging connections. However, sometimes it is impossible to avoid them. In such a case, it is possible to solder jumper wires between the portions of the circuit that are to be connected. This is especially undesirable for long runs, and it is better avoided even for short runs. A better solution is to use PCBs that have more than one conducting layer. The simplest of these is a two-sided PCB. This is a board that has copper laminated to both sides. Separate patterns are etched on the two sides to effect the desired circuit layout and cross-connections. However, in order to make such cross-connections, and to complete the connection between the two sides of the board, it is normally necessary to use epoxy fiberglass because of the necessity of plating holes through the board to connect the top layer to the bottom layer. Holes that are drilled in an SRBP board are adequate to support the leads of components that are placed in the board for soldering, but they are not normally clean enough when drilled or punched to permit satisfactory electroplating of connections between layers of the board. There is thus a long-felt need in the PCB industry for a way of making PCBs with crossover connections on single-sided SRBP boards without using soldered jumper wires.

The same considerations apply equally as much to boards having more than two layers. These boards, referred to as multilayer boards, are often used in more complicated circuits where one set of bridging connections is not enough. As with the two-sided board, it is normally necessary to use epoxy fiberglass for such PCBs. It can be seen that the result of a need for crossed connections in the typical etched copper PCB leads to an increased cost because of the need for epoxy fiberglass in the PCB instead of the less-expensive SRBP boards.

The problems just described become extreme in the case of PCBs for keyboards. A keyboard for a typewriter, computer or the like typically generates an electrical signal when a key is depressed to make an electrical contact or an inductive or capacitive coupling. Such a coupling is made directly or indirectly between two separated conductors on the board or on a flexible plastic membrane that is spaced apart from the board and makes a conductive or field contact when pressed toward the board. When the keyboard is the typical typewriter or computer keyboard, it is impossible or nearly impossible to avoid crossed connections. The physical size of the hands of an operator also sets a limit to the minimum size of a PCB for a keyboard, since it is necessary to place a keyboard switch on the PCB at a location directly beneath the key to be depressed. As a result,

the typical PCB for a keyboard is of the order of ten to eighteen inches by four to seven inches (25-45 cm. by 10-18 cm.). This size requirement has caused the PCB to become a significant part of the cost of a typewriter or computer keyboard. It is not normally possible to use an SRBP PCB because of the need for crossovers and the attendant plated connecting holes. The result is a relatively expensive epoxy fiberglass PCB, laminated and etched on both sides. This cost could be greatly reduced if it were possible to use an SRBP board that contained a circuit on only one side.

A second problem in the manufacture of PCBs for keyboards is the fact that keyboards either have pairs of exposed electrical conductors that are bridged by another conductor or coupled capacitively or inductively to make an electrical connection when a key is depressed, or else have flexible membranes that couple to the board when pressed. It is necessary to apply some form of protection to the exposed electrical conductor so as to minimize the buildup of corrosion that would interfere with the making of the electrical connection. This is most commonly done by etching a copper pattern of interlaced combs, parallel conductors or the like and plating gold to the combs to provide a contact surface that is conductive electrically and that is not readily corroded by exposure to the atmosphere. Gold may be plated to the copper either by electroplating or by electroless plating. Either of

these represents an additional element that contributes to the cost of preparing a PCB for a keyboard.

The usual intended use of a PCB is to serve as a mount for components such as resistors, capacitors, diodes and transistors. Any of these components is typically inserted by placing its leads into holes in the PCB which is then passed through a wave-soldering process to attach the components physically and electrically to the PCB. During the process of manufacturing the PCB, the board is typically coated in part with an organic polymer solder resist to prevent solder from adhering to the covered regions. If the PCB is one designed for a keyboard, the resist is typically deposited so as to cover conductors on the keyboard surface but is masked to leave the comb or other keyboard switches exposed for operation.

A use that is analogous to that of PCBs is the manufacture of electrical cables or the like by depositing conducting material on flexible plastics such as mylar (R.T.M). This is often done by some form of printing process such as screen printing . Most, if not all, of the materials, typically plastics, that are used for flexible cables or flexible flat conductors are not adapted for wave-soldering, and it is therefore necessary to make compression connections or the like at the ends of the cable or flat conductor. As a result, there is no way to attach resistors or capacitors to flexible material, and a rigid PCB is

therefore used with the flexible material to serve as a component mount.

SUMMARY OF THE INVENTION

It is an object of the present invention to provide an alternative method of making electrical interconnections..

It is a further object of the present invention to provide a better way of making printed circuits.

It is a further object of the present invention to provide a better way of making printed-circuit boards..

It is a further object of the present invention to provide a method of making crossed electrical connections on a single side of printed-circuit board.

Other objects will become apparent in the course of a detailed description of the invention.

Printed circuits are produced by screen printing or the like to deposit conductors, resistors, capacitors and insulators. Crossover connections are made by covering the conducting portions to be crossed with an insulating material that serves as a base for a printed crossing conductor. Exposed conductors, as for keyboards and compression connections, are protected from the development of high-resistance corrosion products by printing or overprinting them with an ink that deposits a carbon layer. The method permits the use of rigid printed-circuit boards having crossed connections without requiring the use of a two-sided board. The equivalent of multilayer boards can be achieved by printing repeated layers of conductors, components and insulators on the same side of the board, or by printing on two sides of a board. Interconnection between adjacent layers can be made as a part of the printing process. Resistors can be printed by screening or otherwise depositing a controlled amount of a resistive ink. Capacitors can be produced by printing conductive layers separated by

insulating layers. The process may produce printed-circuit boards containing resistors and capacitors without the necessity for inserting and wave-soldering components. If such a board is to contain components in addition to those printed by the process of the present invention, a printed carbon layer can be used as a solder resist to protect printed conductors during the wave-soldering process, and to protect exposed contacts, as for keyboards, during wave-soldering and in use. The process also permits the printing of a conducting layer as a shield against radio-frequency interference or as a ground plane to provide electrical isolation of portions of a circuit from each other.

A method in accordance with the invention permits production of a printed circuit on a flexible substrate. Resistors, capacitors, conductors and insulators may be produced on a substantially planar solid surface or on a curved surface.

DETAILED DESCRIPTION OF THE INVENTION

The examples that follow represent particular applications of the invention. In the case of those applications to keyboards for typewriters or computers or the like, the scale is fixed by the size of the hands of an operator, and the resulting necessity to separate the keys physically. These dimensions fix the size of such a circuit board. Some of the examples

relate to conventional printed-circuit boards in which components will be inserted and wave-soldered. The possibility of limiting the printing to one side of a board also makes it possible to print a circuit on any surface on which it can be printed and on any substance that is compatible with the printing process. Thus, the plastic of a car dashboard, the case of a radio, a flexible plastic or paper transfer medium, or a heat sink could be used as a substrate on which to print a circuit according to the present invention. The examples shown here were also produced by a screen printing process, which represents a preferred method, but it is clear that any method of printing that will handle the conducting, resistive, carbon and solder-resist inks will be adaptable for the present process and for products produced by the process. The other possibilities for doing this include transfer printing, lithography, air brush, hand brush, and the like. The examples that included the printing of resistors and capacitors produced such components having values that were repeatable within tolerance ranges of five to ten per cent. These values are thus comparable to those achieved with ordinary discrete components.

Example 1

An SRBP board approximately 14 inches by 4 inches by 1/16 inch contained a laminated layer of copper that

had been etched to produce conducting paths and appropriately placed comb connectors to be bridged by conducting pills when a key was depressed. The combs were plated with gold. In order to function properly, this board needed connections that would cross certain conductors without making electrical contact with them. The crossing connections were achieved by screen printing with an ink containing a polymer resist at the crossings. The resist was cured using ultraviolet radiation. A conducting path connecting the desired points was then printed by a screen process over the cured resist, and the printed conductor was then cured at an elevated temperature. This produced a functioning printed-circuit board for a keyboard that was printed on only one side of the PCB and that had no bridging wire connections.

Example 2

An SRBP board approximately 14 inches by 7 inches by 1/16 inch had been laminated with copper and the copper had been etched to leave an appropriate pattern of electrical connections for a keyboard. This pattern was placed on a single side of the phenolic board. Each of the contact pads for the keys was formed by a screen printing of a conductive ink. This ink was then cured by heating. The contact pads and also exposed conductors for compression connectors at an edge of the board were then covered by screen printing with an ink

containing carbon, and the ink was cured by heating it. A screen printing was then made to apply an insulating polymer resist to all areas of the board except the exposed conductors for the key pads and for compression connections at an edge of the board and at those areas where components were to be inserted and wave-soldered. The cured carbon ink functioned as a solder resist that protected the contact pads from solder during the wave-soldering process. If the board had been subjected to hand soldering, the carbon ink would have protected the contact pads from damage by heat.

Example 3

A keyboard of FR-4 epoxy fiberglass having dimensions of approximately 14 inches by 7 inches by 1/16 inch had been laminated and etched on one side to leave copper conductors in a pattern appropriate for a keyboard. Layers of insulating resist were screened in desired crossover patterns on the side of the board that carried the copper and were cured by ultraviolet radiation. A screen printing was then made with a conducting ink to apply conducting strips on the crossovers and also print the contact areas for keys. This was cured by heating. The areas of screened conductors were then subjected to a further screening process to cover the screened conductors with ink containing carbon. This was then cured by heat. The

board was then equipped with resistors by screen printing with resistive ink in desired locations. After the printed resistors were cured by heat, a protective layer of solder resist was applied and was then cured by ultraviolet radiation. The solder resist left open areas for the application of solder. Because of the carbon covering, this board could be subjected to flow soldering without damaging the screened connectors and components.

Example 4

An SRBP board approximately 1 inch by 2 inches by 1/16 inch had been laminated with copper foil on one side, and the foil was etched to leave a desired circuit pattern. A portion of the board was covered with a screened resist which was then cured by ultraviolet light. A conducting ink was printed by a screen process over the resist to make a conducting crossover. The conducting ink was also placed over an area approximately 1/4 inch square to form one plate of a capacitor. The conducting ink was cured by heat. Resistors were deposited in desired locations by screening resistive ink which was then cured by heat. A layer of resist was deposited over the conducting area to provide a dielectric material for the capacitor. This was then cured by exposure to ultraviolet light. A second conducting layer was printed by screening to cover the resist and form the

second plate of the capacitor and also to connect the capacitor at a desired point on the copper lamination, and the conducting ink was cured by heating it. The result was a circuit board with connections and components. A protective layer of solder resist was applied with holes for solder points, and the resist was cured by ultraviolet radiation.

Example 5

An SRBP board of dimensions approximately 2 inches by 1 inch by 1/16 inch received a screened pattern of conducting ink which was then cured by heat. A pattern of screen resistive inks was placed by a screen printing process in desired locations, and the resistive ink was cured by heat. Selected locations of the screened conductor were then plated with copper in an electroless plating tank. A protective resist was then screened over all but those portions of the board that were to be exposed for contact. This included the areas that were given the copper plate, which readied that region of the board for soldered contact, either manual or automatic soldering. As an alternative, nickel could equally as well have been plated to provide an appropriate soldering surface. The conductors formed by conducting ink will not generally withstand the heat associated with hand soldering, although they may stand up under wave-soldering.

Example 6

An SRBP board 14 inches by 7 inches by 1/16 inch was screen-printed with a conducting ink in a pattern appropriate to a keyboard. The ink was cured by heat, and crossing connectors were placed by the process described above. Resistive ink in controlled patterns was printed and cured by heat to produce desired resistors as described above. A layer of solder resist was then screened on the board in a pattern that left substantially square openings at the locations of the typewriter keys. The resist was cured by exposure to ultraviolet light. The cured resist served as a mechanical spacer for a membrane with appropriately placed conductors that was laid over the resist. This provided a method of making a membrane-switched keyboard without the necessity of tooling to cut openings in a piece of plastic to space the membrane from its mating electrical connection. The connections made here were capacitive, but they could equally as well have been conductive or inductive. The process could also have been applied to make switches using two parallel membranes that were spaced apart by screened and cured resist rather than using a rigid board as one connector support.

All of the boards described in the examples above have in common the fact that all of their printing is done on one side of the board. Some boards have been made in the past with jumper wires that are

wave-soldered as a part of the process of making the board, but this is seldom a good solution to the problem of bridging connectors and it is often unfeasible if the run of conductors to be bridged is of any length. Another problem that can be overcome much more simply than the present practice is that of providing shielding against radio-frequency interference (RFI shielding). One reason for the use of multilayer boards is to place a shielding layer protecting portions of the circuit. This is done much more simply on a single layer board on one side by depositing and curing a conducting layer over a layer of resist.

The process of the present invention is adaptable to print as many as thirty to fifty layers, one on top of the other. The practical minimum spacing between adjacent conductors is of the order of 0.01 inch. A screened layer after curing can typically be controlled in thickness to plus or minus 5 microns. Capacitors of various values ranging up to 1000 picofarads may be made as described in the examples, with tolerances to 5 per cent.

In some circumstances, pinholes in one of the applied layers may cause problems, especially in an insulation layer where pinholes may cause unintentional and catastrophic connection between conducting layers. In order to reduce the risks of this problem arising, the or each insulating layer may be applied in two

stages, so that two coatings are applied with variation of coating direction or slightly positional variation between the coatings. For example, when applying an insulating layer using screen printing the first stage comprises applying a screened coating with the direction of application along the X-axis of the screen and work-piece. The second stage comprises applying a further screened coating with a variation from the original coating, for example either by rotating <u>both</u> the screen and the work-piece through 90°, relative to the coating direction, and/or with the screen offset relative to the work-piece, e.g. in the X and/or Y direction, by a minimal amount e.g. a small part of a millimetre.

The Table is a listing of the inks that were used in a screen printing process to produce the examples above.

TABLE

SCREEN PRINTING PROCESS INKS

1. Resist. This is a cross-linking polymer sold under the trademark "Photocoat." The examples used type 2G which is not flexible, although a style 3G is available that is flexible. The examples used a resist that was curable by ultraviolet radiation. A heat-curable resist could equally as well have been used.

2. Resistive Ink. All of the examples used a polymer thick-film ink sold under the trade name "Matthey Lec" R-4000 series. This is available in a range of resistivities and is cured by heat.

3. Carbon Ink. This is a polymer thick-film ink containing carbon black. It is cured by exposure to heat.

4. Conductive Ink. This is a polymer thick-film ink containing silver flakes. It is cured by exposure to heat.

THE CLAIMS

What we claim is:

1. A method of effecting an electrical connection between a first electrical conductor that is affixed to a surface and a second conductor that is affixed to the surface, the connection crossing a third electrical conductor that is affixed to the surface without making electrical contact with the third conductor, the method comprising the steps of:

   (a) placing an insulating polymer on a desired path that bridges the third conductor;

   (b) curing the polymer;

   (c) placing a conducting layer on the desired path between the first and second conductors and on the resist; and

   (d) curing the deposited conducting material.

2. The method of Claim 1 wherein the surface is a printed-circuit board and wherein the first, second and third electrical conductors are made of copper formed by laminating copperfoil to the board and etching to remove copper from portions of the board.

3. The method of Claim 1 wherein the surface is a printed-circuit board and wherein the first, second and third electrical conductors

are made of a conducting ink that is printed in a desired pattern on the board and cured.

4. The method of Claim 1 wherein the step of placing the insulating polymer comprises the steps of printing an insulating ink in the desired path by a screen printing process and curing the ink.

5. The method of Claim 1 wherein the step of placing the insulating polymer comprises the steps of printing an insulating ink in the desired path by a transfer printing process and curing the ink.

6. The method of Claim 1 wherein the step of placing the insulating polymer comprises the steps of printing an insulating ink in the desired path by a lithographic printing process and curing the ink.

7. The method of Claim 1 wherein the step of placing the conductive layer comprises the steps of printing a conductive ink in the desired path by a screen printing process and curing the ink.

8. The method of Claim 1 wherein the step of placing the conductive layer comprises the steps of printing a conductive ink in the desired path by a transfer printing process and curing the ink.

9. The method of Claim 1 wherein the step of placing the conductive layer comprises the steps of printing a conductive ink in the desired path by a lithographic printing process and curing the ink.

10. The method of claim 1 comprising in addition the steps of:

(a) depositing resistive ink in a desired pattern on the surface to serve as a circuit resistor; and

(b) curing the resistive ink.

11. The method of Claim 10 wherein the step of depositing comprises the steps of screen printing the resistive ink in the desired pattern and curing the ink.

12. The method of Claim 10 wherein the step of depositing comprises the steps of transfer printing the resistive ink in the desired pattern and curing the ink.

13. The method of claim 10 wherein the step of depositing comprises the steps of lithographing the resistive ink in the desired pattern and curing the ink.

14. A printed-circuit board made by the process of Claim 4.

15. A printed-circuit board for a keyboard made by the process of Claim 4.

16. A printed-circuit board made by the process of Claim 11.

17. A printed-circuit board for a keyboard made by the process of Claim 11.

18. A method of producing a contact pad for a printed-circuit board for a keyboard comprising the steps of:

(a) printing a pattern of the contact pad in a conductive ink in a desired location on the printed-circuit board;

(b) curing the conductive ink;

(c) printing the pattern in a carbon ink over the cured conductive ink; and

(d) curing the carbon ink.

19. The method of Claim 18 wherein the steps of printing comprise screen printing the conductive ink and screen printing the carbon ink.

20. A method of producing a capacitor by screen printing on a surface comprising the steps of:

(a) printing a first layer of conducting ink on the surface in a predetermined pattern;

(b) curing the conducting ink;

(c) printing an insulating ink in a pattern that substantially covers the first layer;

(d) curing the insulating ink;

(e) printing a second layer of conducting ink on the cured insulating ink in a pattern that is spaced so as to be close to the first layer while avoiding electrical contact with the first layer; and

(f) curing the second layer.

21. A method according to any one of the claims 1 to 13 and 18 to 20 wherein at least one of the layers of insulating polymer is applied in two stages.

22. A method according to claim 21 wherein the first stage comprises applying a screened coating layer in a coating direction, rotating both screen and the surface on which the layer is to be applied through 90° relative to the coating direction and applying a further screened coating layer in the same coating direction.

23. A method according to claim 21 wherein the first stage comprises applying a screened coating layer in a coating direction, offsetting the screen relative to the surface on which the layer is to be applied in the coating direction and/or at 90° to the coating direction by a minimal amount and

applying a further screened coating layer in the coating direction.

24. A printed circuit produced on a single side of a substrate comprising a first set of electrical conductors carried on a surface of the substrate, a layer of an insulating polymer applied on a portion of said surface of the substrate and bridging a first of said electrical conductors on the surface, and bridging electrical conductor carried on the surface of said layer of insulating polymer opposite said surface of the substrate so that it is insulated from said first electrical conductor and in electrical contact with at least a second and a third of said electrical conductors on said surface at opposite sides of siad first electrical conductor.

25. A printed circuit according to claim 24 comprising a second layer of insulating polymer applied on portions of said surface but which leaves exposed parts of said first set of electrical conductors, and a second set of electrical conductors carried partly on said second layer of insulating polymer but making electrical contact with some of said first set of electrical conductors at said exposed portions.

26. A printed circuit according to claim 25 comprising at least one further layer of insulating polymer applied on said surface and/or on parts at least of previously aplied insulating layers and conductors, which leaves exposed parts of at least one set of previously applied electrical conductors.

27. A printed circuit comprising at least one set of electrical conductors carried by a substrate including a first conducting area connected to a first electrical conductor to provide one plate of a capacitor, a layer of dielectric material applied over said area and a second conducting area carried by said layer to form a second plate of the capacitor, and connected to a second electrical conductor.

28. A printed circuit according to claim 27 wherein said second electrical conductor belongs to the same set as said first electrical conductor.

29. A printed circuit according to claim 27 comprising a first set of electrical conductors including said first electrical conductor a first layer of insulating polymer including said layer of dielectric material applied to said substrate and to parts at least of said first set of electrical

conductors and a second set of electrical

Amendments to the claims
have been filed as follows

29. A printed circuit according to claim 27 comprising a first set of electrical conductors including said first electrical conductor a first layer of insulating polymer including said layer of dielectric material applied to said substrate and to parts at least of said first set of electrical conductors and a second set of electrical conductor carried by said substrate and said first layer.

Published 1990 at The Patent Office, State House, 66/71 High Holborn, London WC1R 4TP. Further copies may be obtained from The Patent Office.
Sales Branch, St Mary Cray, Orp ... St Mary Cray, Kent, Con 1/87